ELSEVIER

Available online at www.sciencedirect.com

SciVerse ScienceDirect

Procedia Engineering 28 (2012) 27 – 32

Procedia Engineering

www.elsevier.com/locate/procedia

2012 International Conference on Modern Hydraulic Engineering

Investigation of Hydraulic Characteristics of a Volute-type Discharge Passage based on CFD

Honggeng Zhu[a], Rentian Zhang[a,b], Guoqiang Luo[c], Bin Zhang[b], a*

[a]*Yangzhou University, 131 M Jiangyang RD, Yangzhou, 225009, P. R. China*
[b]*Jiangsu Surveying & Design Institute of Water Resources Co., Ltd., 131 M Jiangyang RD, Yangzhou, 225009, P. R. China*
[c]*Taihu Region Water Project Administration Department, 24 East Taihu RD, Suzhou, 215128, P. R. China*

Abstract

The internal flow patterns of a volute-type discharge passage were investigated in a mixed-flow pumping system based on CFD by using the RNG $\kappa-\varepsilon$ turbulence model to close the time-averaged Navier-Stokes equations. According to the numerical analysis of the internal flow, the hydraulic characteristics of the volute-type discharge passage and the velocity distribution features in several key cross sections were analyzed and quantitatively calculated in terms of velocity distribution uniformity and bias angle. The computed results show that the internal flow field is very complicated involving flow separation and vortex in some regions and the velocity distribution is uneven. The uniformity curves of axial velocities in the outlet cross-section of the volute and the outlet cross-section of the passage are protruding ones while the bias angle curves in the same cross-sections are concave ones, both of them reaching their maximum or minimum value respectively at the flow rate corresponding to the best efficiency point of the pumping system. The hydraulic loss of the volute-type discharge passage varies with the discharge and reaches its minimum when the pumping system efficiency approaches its highest. When the pumping system operates under off-design condition on the large flow rate side, the hydraulic loss of the passage tends to increase rapidly causing the pumping system efficiency to drop sharply. Therefore, appropriate regulation methods should be adopted to adjust the pump operating condition to ensure the pumping system working within high efficiency zone.

Keywords: Hydraulic characteristics; pumping system; volut-type discharge passgae; numerical analysis; CFD

1. Introduction

* Corresponding author. Tel.: +0-086-514-8797-9532; fax: 0-086-514-8797-8606.
E-mail address: zhuhg@yzu.edu.cn.

 doi:10.1016/j.proeng.2012.01.678

A discharge passage constitutes an important part of a low-head large pumping system, the function of which is to enable water to diffuse gradually and change direction easily in the process flowing to the discharge pool, to turn the kinetic energy of flowing water obtained from the pump impeller into pressure energy, and to recover energy farthest without flow separation. The common discharge passages include siphon type, straight pipe type and inclined type as well, among which the siphon-type and straight-type are more widely used in pumping stations as shown in Ref. [1]. The volute-type discharge passage discussed in this paper can effectively decrease the passage height. By means of open chamber design and installing small flap valves on the fast gate, it can adapt large range of water level fluctuation of discharge pool to ensure safety operation of pumping sets as indicated by Zhou [2].

2. Numerical simulation methods

2.1. Governing equations and turbulence model

The configuration of the volute-type discharge passage is shown in Fig. 1, consisting of a volute casing, diffusing segment, and an outlet segment. As Cao et al [3] and Zhou et al [4] discussed that when a pumping system is operating steadily and its internal flow is three-dimensional incompressible viscous flow, it can be described by the mass conservation equation and the time-averaged N-S equations.

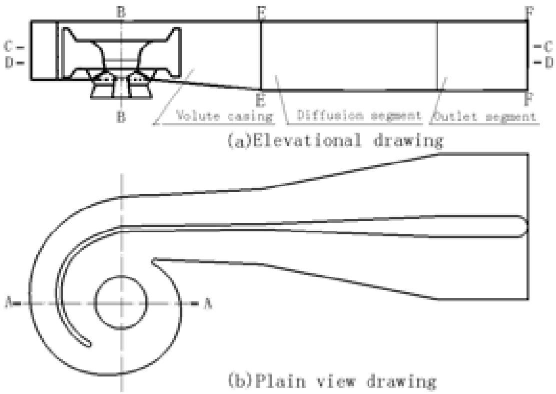


Fig. 1. A volute-type discharge passage

The RNG $\kappa-\varepsilon$ turbulence model is adopted to close the N-S equations in the numerial analysis. As Yakhot et al. [5] decribed that the RNG $\kappa-\varepsilon$ turbulence model takes the same form as standard $\kappa-\varepsilon$ turbulence model, but the coefficients in the model are completely obtained through theoretic analysis. In the laminar layer the turbulence viscosity coefficient is equal to the molecule viscosity, and beyond the layer the turbulence viscosity increases very rapidly, so the RNG $\kappa-\varepsilon$ turbulence model can handle ideally flow fields with flow separation and backflow.

2.2. Grid generation, discretization and solution

The whole computation domain includes a sump, a bell-type suction box, a mixed-flow impeller, a volute-type discharge passage and a discharge pool. Six-face-body structured grids are generated for the sump and discharge pool to reduce the node number and computation cost. The suction box, discharge passage and impeller adopt four-face-body unstructured grids to accommodate complicated shapes of impeller and flow passages. Grid generation was accomplished by means of commercial code Gambit. The total number of calculation nodes is about 900,000. The mesh of the computed pumping system is shown in Fig. 2. The finite volume method is used to discrete the flow domain and arithmetic SIMPLEC is adopted to couple pressure and velocity fields.

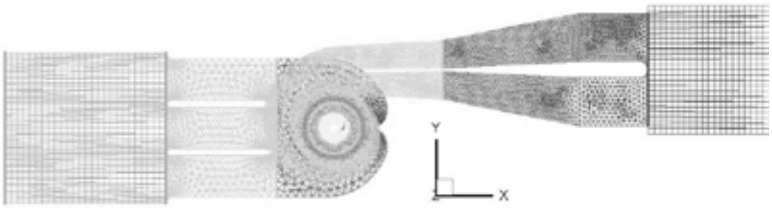
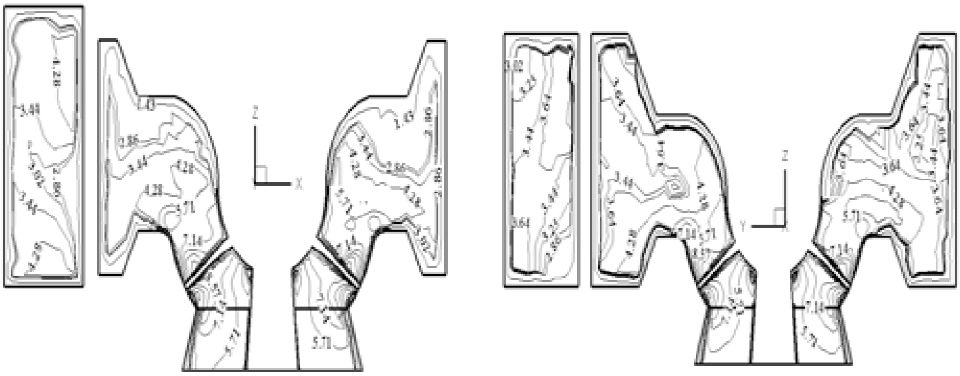


Fig. 2. Mesh of the computed domain

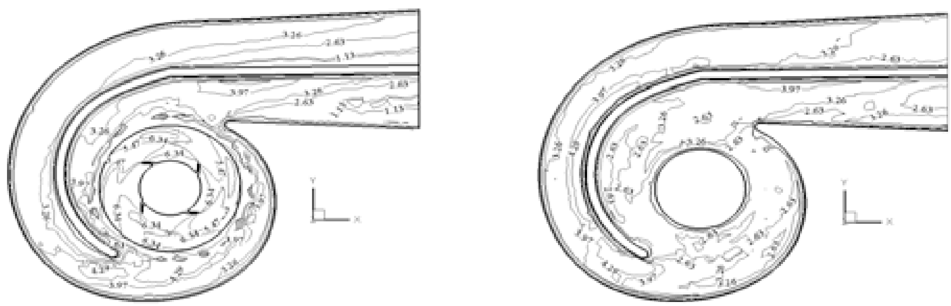
3. Analysis and discussion of computed results

3.1. Basic flow patterns of volute-type discharge passage

The internal flow patterns of a volute-type discharge passage are closely related with its design parameters and the performances of pump. Numerical analysis shows the internal flow fields in its typical cross sections and the velocity gradients are quite high in some regions even the pumping system is operating near the best efficiency point.



(a) A-A (b) B-B



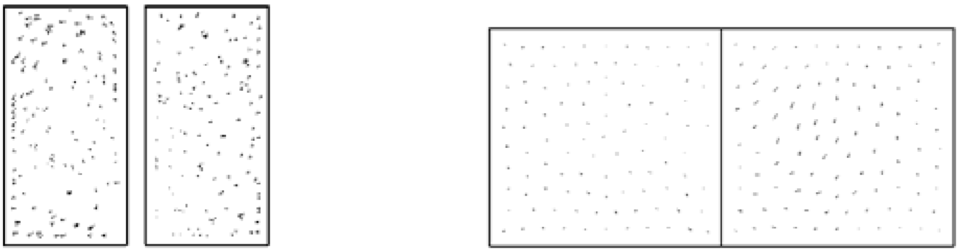
(c) C-C (d) D-D

Fig. 3. Contours of velocity field (m/s)

Fig. 3 shows that water flowing into the inlet pump is basically symmetric in the Y direction. Water runs out of the impeller around the volute. As the cross-sectional area increases the velocity of flowing water slows down, and part of is kinetic energy is converted into pressure energy. After passing the diffusion segment the velocity of water decreases further, and finally flow into the discharge pool. In Z direction, because the D-D section is closer to the exit of the impeller, while the C-C section has a certain distance, so there is big difference in velocity distribution of the two sections. The latter has a much more uniform distribution. From the velocity contours of C-C and D-D sections, it can be obviously seen that when water goes around the volute, affected by centrifugal force, water is thrown off and the outside velocity of the volute is obviously higher than the inside velocity. Furthermore, from the velocity contours in C-C and D-D sections, low velocity backflow wakes can be clearly observed when water flows round the partition tongue of volute and separating wall.

3.2. Velocity distribution features in the outlet sections of the volute and the passage

The flow fields in the outlet sections of the volute and the passage are shown in Fig. 4. From Fig. 4(a) vortex in the volute outlet can be obviously seen. Even being regulated after water passing through the diffusion segment and outlet segment to the outlet of the passage the velocity distribution is not symmetric, uniform and axial on both sides (see Fig. 4(b)). There exists transverse velocity and the phenomena of secondary flow are visible. The design of a volute will have reflections in the bias angle of the outflow and axial velocity distribution uniformity. The so-called bias angle refers to the deviation extent of actual velocity of a section from the direction of axial velocity. The value of bias angle reflects the magnitude of transverse velocity, so the smaller it is the better. With regard to axial velocity distribution uniformity the higher the better flow patterns is obtained. The different design schemes of diffusion and outlet segments of a discharge passage can be compared and evaluated to guide hydraulic design optimization in terms of distribution uniformity of axial velocity and bias angle of the outflow.



(a) E—E (b) F—F

Fig. 4. Velocity vector profile in outlet sections of the volute casing and passage

The velocity distribution characteristic curves in the volute and passage outlet sections are shown in Fig. 5, from which it can be seen that the values of velocity distribution uniformity vary with the flow rate and the characteristic curves are protruding ones, the maximum values of both curves not exceeding 45%. As the increase of flow rate of the pumping system, the values of uniformity increase, and after reaching their crests, they go down slowly. The correspondence of maximum values of velocity distribution uniformity curves with high efficiency region of the pumping system shows that the pump performance affects the internal flow characteristics of the passage, so that the research of discharge passage should not be separated from pump performance. The bias angle curves in the outlet sections of the volute and passage are concave ones. Along with the increase of flow rate, the averaged bias angles in both sections decrease gradually, and attain their minimums when the pumping system efficiency reaches its maximum. Afterwards, The bias angles increase with the increase of flow rates slowly.

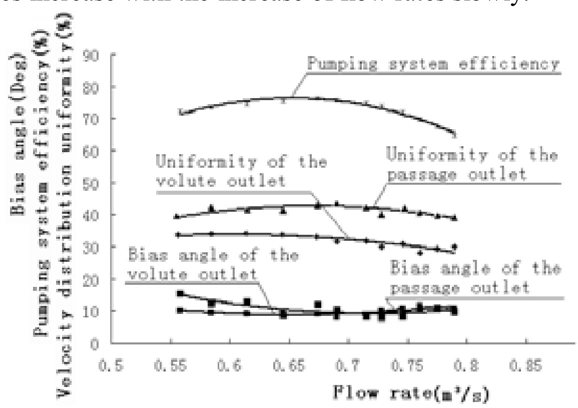


Fig. 5. Velocity distribution characteristic curves

3.3. Calculation of head loss

The head loss curve of the volute-type discharge passage is shown in Fig. 6. For the purpose of analysis, the total head loss of the passage is divided into two parts, namely, the head loss of volute segment and the head loss of diffusion and outlet segment. Fig. 6 indicates that the head loss of volute varies with flow rate and it is a concave curve opening upwards, reaching its minimum when the pumping system efficiency approaching its highest point. The head loss of diffusion and outlet segment, relatively smaller, increases with the increase of flow rate and is similar in shape with the head loss curve of the volute. The total head loss of the passage is the summation of the two parts mentioned above, possessed the same feature as the volute head loss.

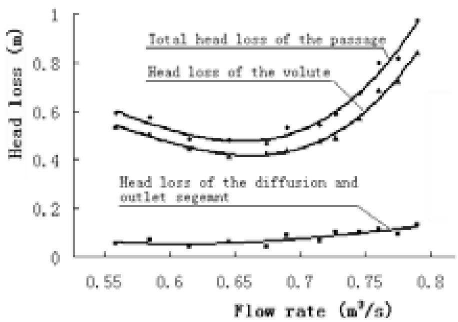


Fig. 6. Curves of head loss with flow rate

Contrasting Fig. 5 with Fig. 6 it can also be seen that when the pumping system operates on off-design conditions, the influence of head loss of volute-type discharge passage on the pumping set efficiency is different. The head loss of the passage increase slowly with the increase of flow rate when the pumping system runs on the left side of the best efficiency point, while on the right side of the best efficiency point, the head loss increases very quickly as the flow rate increases. For instance, the highest efficiency of the pumping system goes as high as 76.3%, the corresponding head loss of the passage is 0.47m, accounting for 5.46% of net lift of the pumping set. If the flow rate is increased by 15%, the head loss of the passage goes up to 0.83m, and the pumping set efficiency decreases to 67.7%. Therefore, in order to keep the pumping set running in high efficiency zone, the variation of flow rates should be limited.

4. Conclusion

The CFD analysis shows that the internal flow pattern of volute-type discharge passage is very complex, there is vortex and flow separation in typical cross-sections. Bias flow is obvious in the outlet sections of the volute and the passage, and velocity distribution is not uniform. The distribution uniformity curves of axial velocity in the outlet sections of the volute and the passage are protruding ones opening upward, and the bias angle curves in these two sections are concave ones opening downward. These uniformity curves and bias angle curves reach their maximum or minimum respectively corresponding to the best efficiency point of the pumping system. The head loss of volute-type discharge passage varies with the flow rate, which reaches its minimum when the pumping set efficiency is at the peak. When the pumping system operates under off-design conditions and on the large flow rate side, the head loss increases rapidly and the pumping set efficiency decreases sharply, hence, appropriate adjuysting methods should be adopted to to realize economic operation of pumping station.

Acknowledgments

This paper is sponsored by the Hydraulic Engineering Project from the Water Resources Department of Jiangsu Province(No.2010023), and the Open Project from the Key Lab of Hydraulic & Power Engineering of Jiangsu Province (No.K100016).

References

[1] Hubei Surveying and Design Institute of Water Resources. Design drawings of small-scale water conservancy and hydropower projects. Beijing: Water Conservancy and Electric Power Press;1983.

[2] Zhou J. Pumping set performance and structure of No.1 Zaohe pumping station. *Jiangsu Water Science and Technology*; 1992, No. 2: p. 33-46, in Chinese.

[3] Cao S, Xu G, Wu Y. Numerical analysis of three-dimensional turbulent flow through a turbine spiral casing for Three Gorge Project. *Journal of Tsinghua University (Sci & Tech)*;1997, Vol. 37, No. 8: p. 95-98, in Chinese.

[4] Zhou X, Qu L, Wu Y. Numerical analysis of internal flows through the spiral casing and stay vanes. *Journal of Tsinghua University (Sci & Tech)*;1997, Vol. 40, No. 8: p. 93-97, in Chinese.

[5] Yakhot V, Orszag SA. Renormalization group analysis turbulence: basic theory. *Scientific Computation*; 1986, No. 1: p. 3-20.